International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 10, December- 2012

Static Structural And Modal Analysis Of Engine Bracket Using Finite Element Anlysis.

Umesh S. Ghorpade, Prof. D.S.Chavan, Prof. M.V.Kavade
Mechanical Engineering Dept.
Rajarambapu Institute of Technology, Sakharale.
Islampur, India.

Abstract

The need for light weight structural materials in automotive applications is increasing as the pressure for improvement in emissions and fuel economy increases. The most effective way of increasing automobile mileage is to reduce vehicle weight. The incorporation of aluminium and magnesium alloys into automotive structures has steadily increased to meet all these requirements. The automotive engine mounting systems are very important due to different aspects of vehicle performance. Engine bracket has been designed as a framework to support engine. Vibration and fatigue of engine bracket has been continuously a concern which may lead to structural failure if the resulting vibration and stresses are severe and excessive. It is a significant study to understand the structural characteristics and its dynamic behavior. This paper presents and focuses on some Finite Element (FE) analysis of a typical engine bracket of a car will be carried out and natural frequency will be determined.

1. Introduction

Introduction of light weight material in automobile industries is increases to improve performance of vehicle. So use of light weight materials such as aluminium and magnesium alloys to increase fuel economy. Today's interest in magnesium alloys for automotive applications is based on the combination of high strength properties and low density. For this reason magnesium alloys are very attractive as structural materials in all applications where weight savings are of great concern. In automotive applications weight reduction will improve the performance of a vehicle by reducing the rolling resistance and energy of acceleration, thus reducing the fuel consumption. Magnesium with its good strength to weight ratio is one of the candidate materials to realise light weight construction, but it has to compete with various other materials. Material selection is thereby determined by economical issues as much as by its component characteristics or properties. However magnesium shows high potential to substitute conventional materials. Magnesium alloys should be used in applications where low mass and high specific properties are required.

Engine is one of the most important components of a road vehicle such as car. High performance sports car has their engine supported by bracket. It plays an important role in improving the comfort & work environment of a car. The improvement of engine bracket system has been the subject of intense interest for many years. It is necessary to design proper engine bracket for a car. As such, engine bracket has been designed as a framework to support engine. Vibrations and fatigue of engine bracket has been continuously a concern which may lead to structural failure if the resulting vibrations and stresses are severe and excessive. Prolonged exposure to whole-body vibration in the working environment may lead to fatigue and in some cases it damages the car. Generally, the most important vibration relevant excitations in a car engine can be identified as follows:- combustion force; main bearing reaction forces including mass forces damper function and flywheel whirling, modified by the front-end damper; piston side forces including secondary motion; camshaft bearing reaction forces including mass forces, opening and closing impacts and bearing impacts; valve opening and closing impacts; valve train forces caused by chain/belt movement or gear drive; gear train forces inside the transmission; drive train

reaction forces and moments. It is well-known from basic Non-linear vibration theory; improvement in the vibration control can be achieved by determining the natural frequency of the engine bracket system well below the frequency band in which excitation exhibits most of the vibratory energy. It is in this context, the development of engine bracket can make the engine capable of absorbing vibration. Automotive engine mounting system must satisfy the primary tasks such as engine movement, engine rigid-body dynamic behaviour, and vibration isolation. The design and development of mounting bracket through use of Ansys software to achieve the requirements for mounting system. Limits over the development of the mounting systems due to drivability and NVH concerns, provides savings in design resources. NVH is an important vehicles characteristic motivating to achieve overall customer satisfaction. Engine is mostly mounted to the front sub frame and once installed in a vehicle, engine mounting has a significant task in decisive the vehicle vibration characteristics. Optimizing the mounts system in early stages of engine design is possible by implementation of computer-aided engineering (CAE) tools. CAE results can be analyzed without any costly prototypes. The results can be used to define strategy for the vehicle mount system and optimize the locations and the rates of mounts. A good mounting system separates engine input vibration from the vehicle body and suppress the effect of road inputs to the vehicle driver.

2. Literature review

Gabriel-Petru ANTON et al.[1] have investigated the NVH test-calculation correlation, the finite element (FE) model updating of an engine and the vibration level (low and medium frequency range) on the engine/body interface points . The main objective for this approach is to obtain the absolutes values of the vibration level (low and medium frequency range) on the interface points using an updated FE model. Experimental and theoretical analysis used for this work, have allowed us to understand the real vibratory behavior and to obtain a new FE model more closed by reality. The final updating, the test-calculation correlation results and also the operational simulation.

S. Irving et al.[2] has investigated the fatigue performance of two different bracket connections for use in high-speed ocean craft. Constant amplitude, cyclic tests revealed that weld quality within the curved or nested insert has a profound effect upon the fatigue behaviour. Under severe conditions, the loss in fatigue performance due to poor weld quality may override the gain achievable by more optimal bracket designs. Under the condition of a good quality butt weld with deep penetration, the nested bracket has an improved fatigue performance when compared to the traditionally used soft toe bracket.

S.K. Loh et al. [3] has discussed about the focuses on some Finite Element (FE) analyses performed such as frequency analysis to determine the structural response due to harmonic excitation over a frequency range. The resonant frequency can be predicted based on the responses in frequency domain. Besides that, the static and dynamic vibration analyses give the maximum structural stress condition under static loading and dynamic condition. The predicted maximum stresses are compared with inherent material yield strength. The plastic deformation is not covered in the study as only elastic property is defined. A fatigue failure prediction of the current P-TAC motor bracket using FE simulation and fatigue failure criteria approach has also been studied. The dynamic stress curve giving mean stress and alternating stress has been applied in the established fatigue failure criteria such as Yield Criteria and Fracture Criteria to predict the possibility of fatigue occurrence. This approach is considered a conservative prediction approach to prevent structural fatigue which is best suited and safe for certain design applications.

Zhang Junhong et al. [4] have investigated that vibratory and acoustic behaviour of the internal combustion engine is a highly complex one, consisting of many components that are subject to loads that vary greatly in magnitude and which operate at wide range of speed. CAE tools development will lead to a significant reduction in the duration of the development period for engine as well as ensure a dramatic increase in product quality. Modern CAE tools allow the analysis, assessment and acoustic optimization of the engine.

3. Proposed Methodology

At first the theoretical study of bracket is done. The overall purpose of engine mounting bracket is to support the Engine and sustain the vibrations caused by engine as well as bumps from tires due to uneven road surfaces. The key areas for modification are identified. The main task in this study is to find the natural frequency of bracket by optimizing it for various material combinations. The 3-Dimensional model is prepared for Bracket. Different types of materials are assigned and analysis is carried out using finite element analysis software named Ansys Inc. Best material is selected for bracket body. The results are compared with practical results.

4. Scope of Work

This paper explains the process of optimization of natural frequency of engine mount bracket. The problem of engine vibration is considered for noise reduction by optimization of natural frequency. Further scope is to use different lightweight materials, which further reduces weight.

5. Objective

To do static structural and modal analysis of engine mounting bracket for three different materials viz. aluminum (Al), Magnesium (Mg) and

Cast Iron (CI) and suggest best material for the bracket.

6. Engine Mounting Bracket

The engine mount assembly includes a support member arranged to be attached to a vehicle frame component. a pair of elastomeric engine isolators positioned relative to each other on the support member, and a pair of engine mounting brackets arranged to be positioned on the isolators and attached to a engine component, where in the engine isolators are arranged to maintain the engine mounting brackets in spaced relation to the support member thereby dampening engine vibration and controlling engine movement relative to the vehicle. An engine mounting system is often a primary path for noise.

Figure 1: Engine Bracket

The vehicle's structure at the mounting location is crucial in regard to noise transmission, durability and crash worthiness. The upper bracket attaches to the transmission by use of two horizontal fasteners. A limited vertical space is available for both the rubber mount and bracket. With limited rubber volume it is not possible to allow the rubber to be as soft as it is desired for maximum isolation of vibration. Therefore the bracket must be designed to be as stiff as possible.

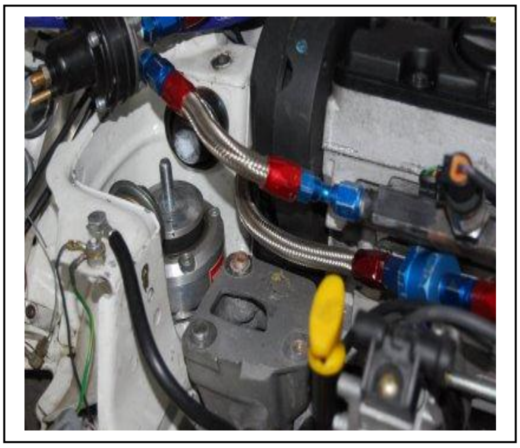

Figure 2: Position of Engine Bracket.

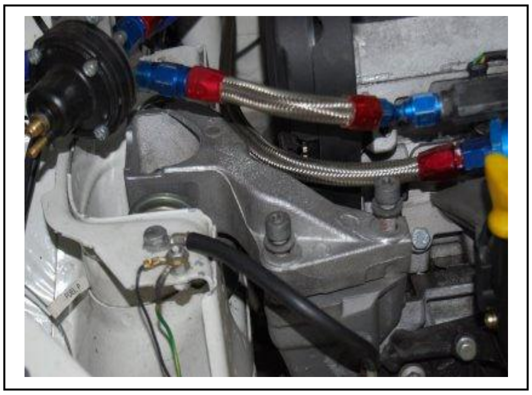

Figure 3: Mounted engine bracket.

7. Analysis of Engine Mounting Bracket

Finite element analysis (FEA) is one of the most popular engineering analysis methods for Non linear problems. FEA requires a finite element mesh as a geometric input. This mesh can be generated directly from a solid model for the detailed part model designed in a three-dimensional (3D) CAD system. Since the detailed solid model (see Fig. 4) is too complex to analyse efficiently, some simplification with an appropriate idealization process including changing material and reducing mesh size in the FE model is needed to reduce the excessive computation time. The engine mounting brackets are made of up Aluminium alloy or Mg alloy or Gray C.I. For thin bodies, a different type of meshing approach is required. For engine mounting bracket part, we extracted and meshed the mid-surface using Hex Dominant Quadrilateral and Triangular elements.

Fig. 4 shows the FEM model of the existing design. The existing design has 4 holes. One hole is fixed and remaining three has force of 1000 N. This force is produced by Thrust. There is also self weight (g). The material used for FE Analysis is Non Linear. The FEM Model having 6 freedoms: translations in the nodal x, y, and z directions and rotations about the nodal x, y, and z-axes.

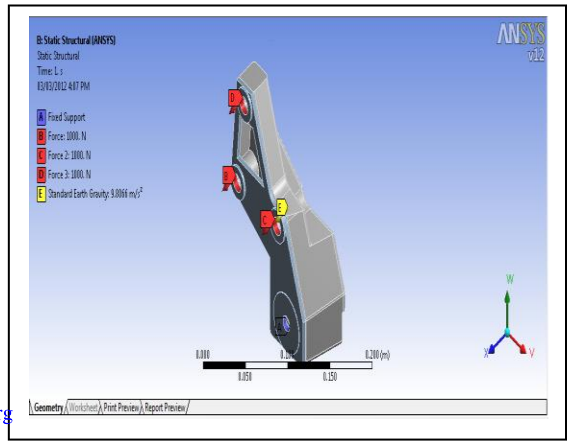

Figure 4: FE Model of Mounted engine bracket.

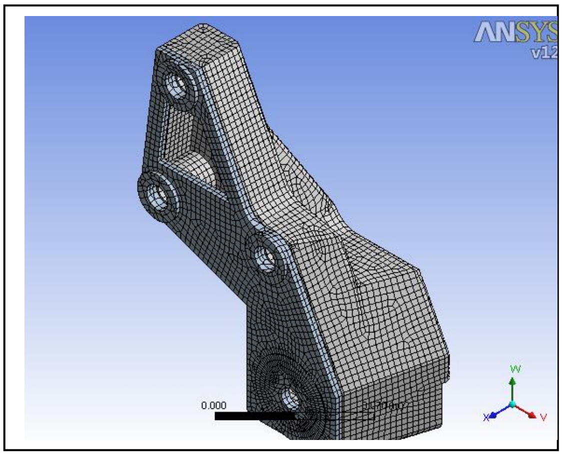

Figure 5: FE Mesh Model of Mounted engine bracket.

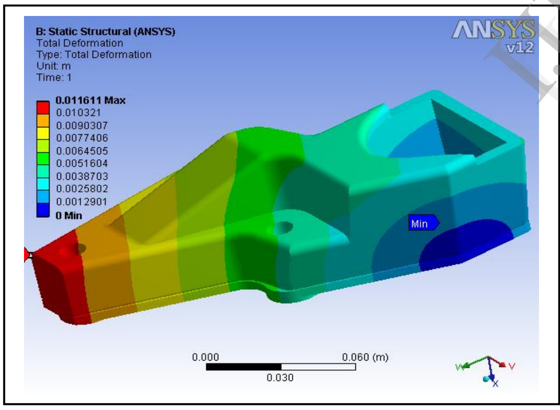

Figure 6: Total Deformation of 2.5 mm Mesh size Aluminium Alloy Mounted engine bracket.

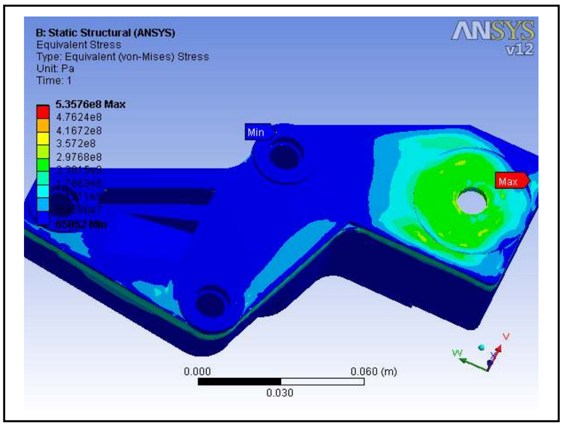

Figure 7: Equivalent (von misses) stresses of 2.5 mm Mesh size Aluminium Alloy Mounted engine bracket.

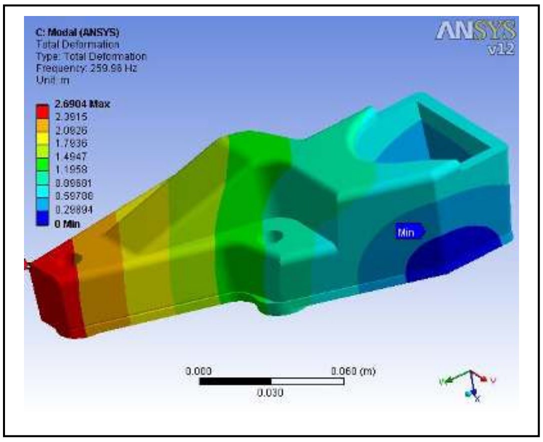

Figure 8: Natural Frequency of 2.5 mm Mesh size Aluminium Alloy Mounted engine bracket.

8. Result and Discussion

Sr. No.	Results			
	Material	*Max. Deformation (M)*	*Max. Stress (Pa)*	*Natural Frequency (Hz)*
1	Al alloy	0.020035	6.1839 x10^{8}	253.28
2	Mg alloy	0.041132	5.1303 x10^{8}	251.31
3	Gray C.I.	0.00434	6.2539x10^{10}	192.77

9. Experimental Test Matrix

The Bracket was tested on FFT Analyser. The vibration signal is sampled by FFT and Accelerometer with Magnetic Base. Excitation methods are used to measure natural frequency of the system. Average natural frequency values (Hz) for AL, CI, and MG are 258.2 Hz, 200.3 Hz and 257.9 Hz respectively.

10. Conclusion

Vibration plays a critical role in Engine components, especially in the supporting bracket. Gray Cast Iron is essentially a brittle material and this is evident in the results that the low natural frequency will prove as a hindrance in vibration characteristic of the bracket.

In terms of analysis, Al alloy and Mg alloy are showing almost same value of natural frequency and indicate that any one of them would be a better choice than Gray Cast Iron.

However, in terms of FEA there is a caveat, which being that in Modal FEA, the effect of Damping is not considered. In Practical terms, Mg alloy exhibits better damping characteristics than Al alloy. Hence as far as the recommendation goes, Mg alloy will be preferred.

11. Acknowledgment

We would like to acknowledge the role of Mr. Vinay Patil, of Vaftsy CAE for his guidance in some aspects of ANSYS solutions.

12. References

[1] Zhang Junhong, Han Jun "CAE process to simulate and optimise engine noise and vibration" Mechanical Systems and Signal Processing 20 (2006) 1400–1409.

[2] Gabriel-Petru ANTON, Mihai PAVAL, Fabien SOREL, "APPLICATION ON AN UPDATED FINITE ELEMENT MODEL OF AN ENGINE IN THE AUTOMOTIVE INDUSTRY" SISOM 2011 and Session of the Commission of Acoustics, Bucharest 25-26 May.

[3] Senthilnathan Subbiah, O.P. Singh, "Effect of muffler mounting bracket designs on durability", Engineering Failure Analysis 18 (2011) 1094–1107.

[4] Youngwoo Choia, Dohyun Jungb, Kyoungchun Hamc, "A study on the accelerated vibration endurance tests for battery fixing bracket in electrically driven vehicles", Procedia Engineering 10 (2011) 851–856.

[5] S.K. Loh a, W.M. Chin a, Waleed F. Faris, "Fatigue analysis of Package Terminal Air Conditioner motor bracket under dynamic loading", Materials and Design 30 (2009) 3206–3216.

[6] S. Irving *, F.Ferguson-Smith, X.Z. Hu, Y. Liu, "Comparative fatigue assessment of soft toe and nested bracket welded aluminium structures", Engineering Failure Analysis 12 (2005) 679–690.

[7] Mehmet,OsmanH.Mete,UmitKocabicak,MuratOzsoy,"Stamping process design using FEA in conjunction with orthogonal regression", Finite Elements in Analysis and Design 46 (2010) 992–1000.

[8] Hong Suk Chang, "A Study on the Analysis Method for Optimizing Mounting Brackets," SAE International, 2006-01-1480.